Layout process:

1. Import netlist, DXF or pdf
2. Collect/build PCB footprints
3. Make board outline and place all symbols into the board
4. Fill Xsection based on stackup and crate artwork film
5. Place symbols based on each group
6. Assign Diff pairs and Run Xnet for all serial Symbols that have length matching
7. Implement constraints: Electrical, Physical, Spacing…
8. Do placement and fanout + place all groups into board
9. Do routing + match length + Cut planes
10. Complete all connection and DRCs
11. Do silk + Assembly
12. Do PCB + QC check
13. Generate gerber package: ODB or Manual
14. Check gerber (use Cam350 or Genesis to check Short/Open) before we send them to Fab/Assy house